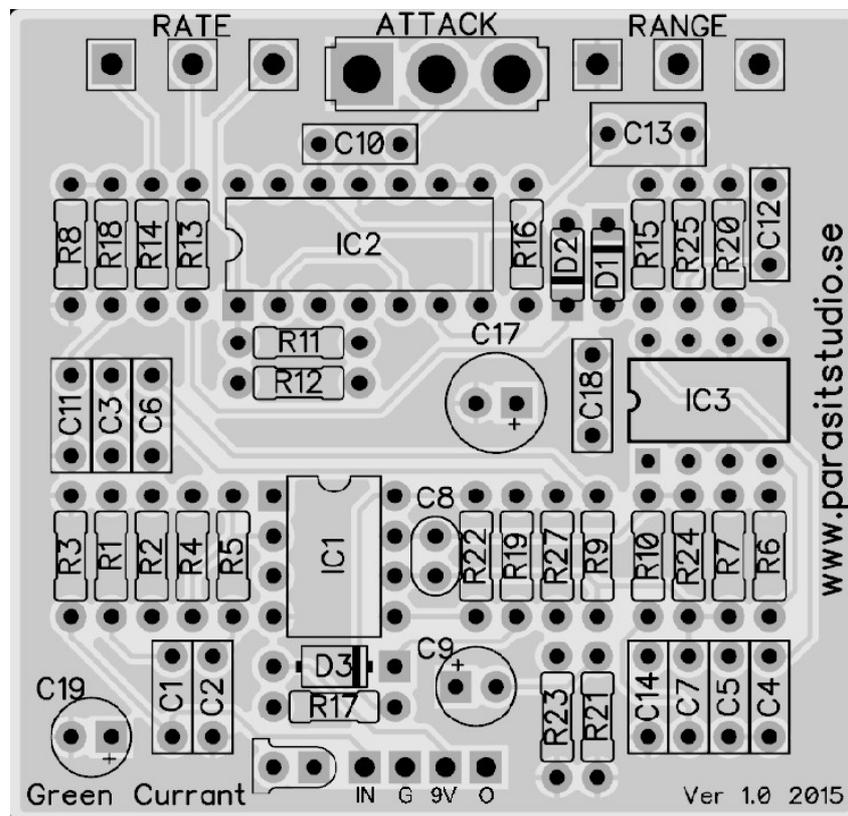


THE GREEN CURRANT TREMOLO

Build Document last updated may 2016

for PCB version 1.0

The Green Currant tremolo is a very percussive and vibey tremolo based around the TDA7052A amplifier chip. It splits the signal into two paths that are out of phase. One side is static and the other is filtered and amplitude modulated, then both paths are summed together. This results in what sounds almost like a light vibrato/phaser'ish effect. This tremolo has a range pot insted of a conventional depth pot. It controls the frequency-range of the modulation. The Attack switch toggles between hard and soft attack (saw/ramp waveshape). Happy playing!



General builds tips

- Solder the low profile components first, from short to tall. Recommended order: resistors, diodes, IC socket, film-caps, electrolytics, pots and switches
- Always use sockets for IC chips and transistors to avoid heating them directly. It also makes it much easier to swap them out if needed.
- Pay special attention to the orientation of the diodes and electrolytics.
- This PCB is designed for board mounted angled pots, but if you want to use regular solderlug-pots, the square holes represents pin 1 of the pot.
- The pots and the switch are meant to be mounted on the backside (solder side) of the board and soldered on the front (component side).
- The LED connections for the rate indicator LED is best to wire offboard to where you prefer it in the box.

Wiring

For more info on how to wire up the stompswitch, jacks ect, please visit the Parasit Studio website and download the PDF called "offboard wiring". You can find it here:

<http://www.parasitstudio.se/build-docs.html>

The Green Currant Bill Of Materials (BOM)

Resistors		Capacitors		IC's	
R1	1K	C1	10nF	IC1	TL072
R2	1M	C2	10nF	IC2	LM324
R3	1M	C3	100nF	IC3	TDA7052A
R4	10K	C4	22nF		
R5	1K	C5	100nF		
R6	47K	C6	15nF		
R7	68K	C7	1nF		
R8	470K	C8	470pF		
R9	100K	C9	4.7uF		
R10	10K	C10	100nF	Potentiometers	
R11	47K	C11	100nF	RATE	B50K
R12	33K	C12	220nF	RANGE	A50K
R13	47K	C13	1uF		
R14	18K	C14	6.8nF		
R15	510K	C17	220uF		
R16	680K	C18	100nF		
R17	4.7K*	C19	47uF		
R18	470K	Diodes		Switches	
R19	47R	D1	1N4148	Attack	SPDT on/on
R20	100K	D2	1N4148		
R21	100K	D3	1N5817		
R22	10K	1x LED			
R23	10K				
R24	150K				
R25	220K				
R27	10K				
CLR**	4.7K – 18K				

BOM Update 2016-05-21

Changed IC2 from a TL074 to a LM324. This will solve an issue with the LFO having different speeds at the two switch positions.

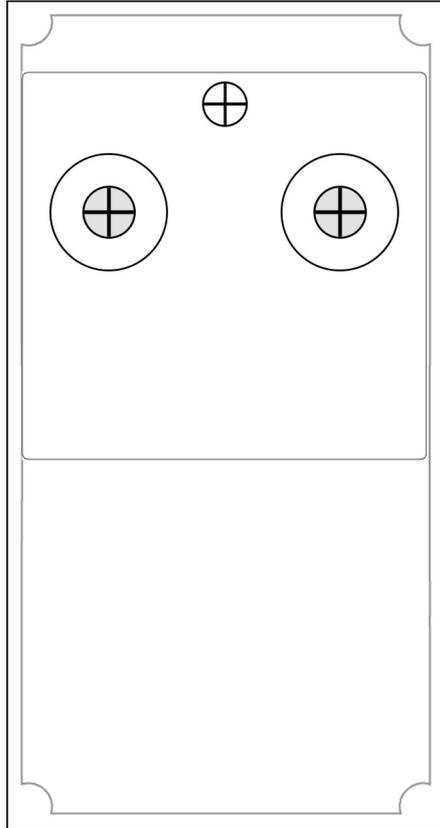
Notes regarding the BOM

- The led symbol next to the input is for the rate indicator LED. The flat side indicates the cathode/negative side (short leg).
- * Current limiting resistor for the rate indicator LED
- ** Current limiting resistor for the bypass LED. It needs to be wired offboard.

Use the appropriate value for the CLR resistors to match the brightness of your LED type. For normal diffused LEDs I recommend using 4.7K resistors and for clear superbright LEDs I recommend using 15K resistors.

- Possible mod: Use a on/off/on switch insted if you want a triangle wave-shape aswell in the middle (off) position. Personally I don't think it sounds good in this circuit, the rate is much slower and it sounds like a panting dog. But it's an easy mod so I thought it would be worth mentioning.
- Also not included in the BOM but good to have: enclosure, input and output jacks, DC jack, 3PDT switch and knobs.

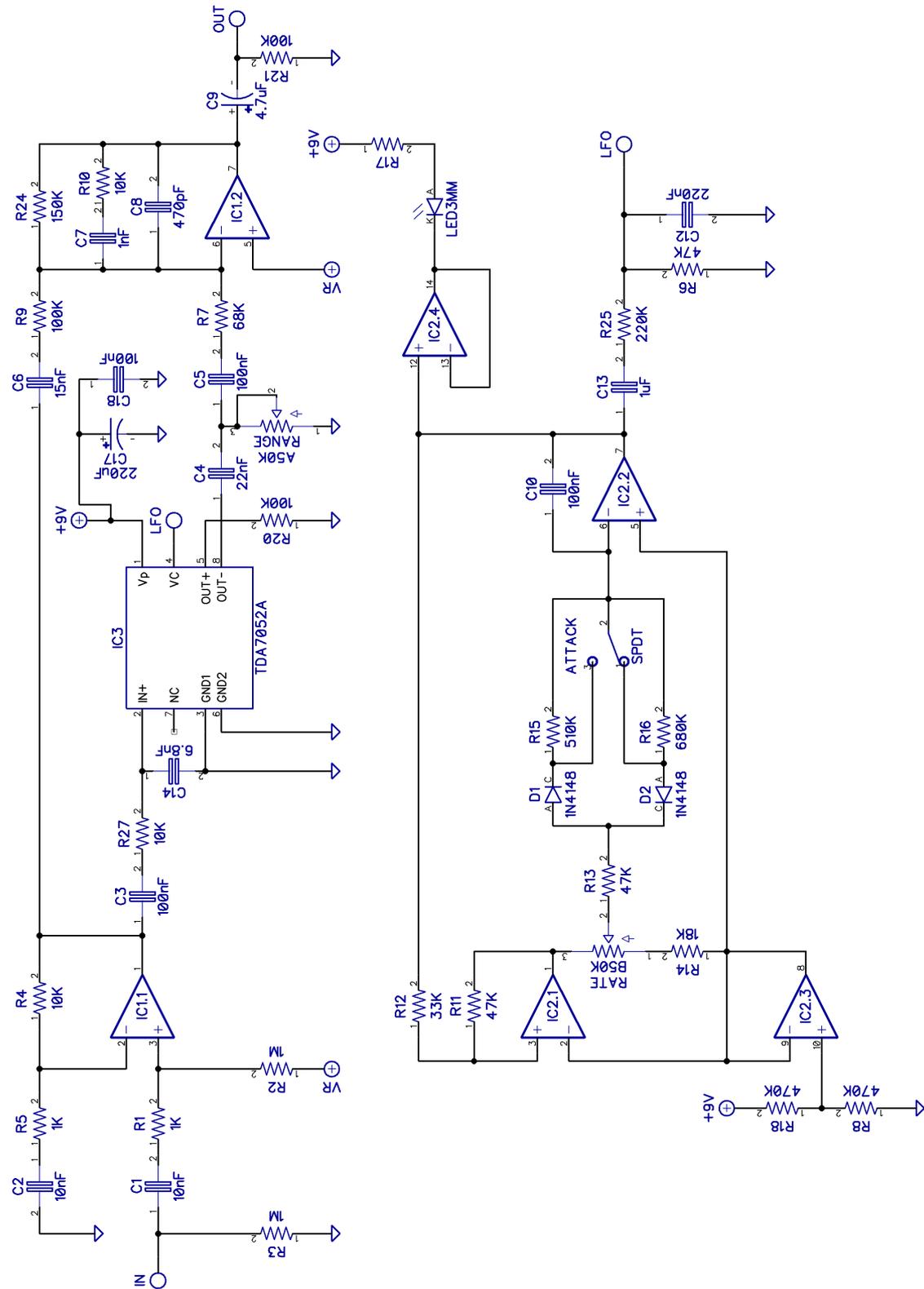
Drilling template (1590B)



Use at your own risk! This template is approximate.

- Make sure your printer isn't doing any scaling / is set to 100% print size.
- Drill stompswitch, DC jack and input/output jack to your own preference.
- This is just a suggestion. If you want the DC jack at the top you should probably have the PCB a bit lower.
- Measure and confirm before drilling!

Schematic



Troubleshooting

There's always a chance of running into trouble. To minimize error, follow the BOM and general building tips carefully. Take your time and don't rush. Take a break now and then. Use good solder, and it helps to have a decent soldering station instead of a cheap iron.

If you are still having trouble, please visit the madbean forum Parasit Studio subforum section and ask for help there.

<http://www.madbeanpedals.com/forum/index.php?board=84.0>

If you have bought the Musikding kit and have received a faulty or missing component, please contact musikding directly.

<https://www.musikding.de/kontakt.php?lang=eng>

Terms of use

PCB's from Parasit Studio are intended for DIY use only. Commercial resale is not allowed. It's meant for personal use, which means that it's not allowed to build a lot of pedals and sell them for profit to strangers using public forums and craigslist ads. However, it's totally ok to build a few pedals and sell to your friends and bandmates. After all, that's what this hobby is about. DIY or DIE!

www.parasitstudio.se
parasitstudio@gmail.com